



Aalborg Universitet

AALBORG UNIVERSITY
DENMARK

User-Dependent CFD Predictions of a Backward-Facing Step Flow

Report to participating teams

Peng, Lei; Nielsen, Peter Vilhelm; Wang, Xiaoxue; Li, Yuguo

Published in:

Proceedings of the 9th International Symposium on Heating, Ventilation and Air Conditioning (ISHVAC) - The 3rd International Conference on Building Energy and Environment (COBEE)

Publication date:
2015

Document Version
Early version, also known as pre-print

[Link to publication from Aalborg University](#)

Citation for published version (APA):

Peng, L., Nielsen, P. V., Wang, X., & Li, Y. (2015). User-Dependent CFD Predictions of a Backward-Facing Step Flow: Report to participating teams. In *Proceedings of the 9th International Symposium on Heating, Ventilation and Air Conditioning (ISHVAC) - The 3rd International Conference on Building Energy and Environment (COBEE)* Tianjin University. <http://www.cobee.org/dct/page/70038>

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal -

Take down policy

If you believe that this document breaches copyright please contact us at vbn@aub.aau.dk providing details, and we will remove access to the work immediately and investigate your claim.

User-Dependent CFD Predictions of a Backward-Facing Step Flow **- Report to participating teams**

Lei Peng¹, Peter V Nielsen², Xiaoxue Wang¹ and Yuguo Li¹

¹Department of Mechanical Engineering
The University of Hong Kong, Hong Kong SAR, China

²Department of Civil Engineering
Aalborg University, Aalborg, Denmark

Version July 6, 2015

ACKNOWLEDGEMENT

We thank all teams and participants for your contribution to the CFD simulations. The response to our call for the workshop has been overwhelming, though the notice was a short one. We are also grateful to the kind support from the conference organizers, Professor Liu Junjie and Professor Qingyan Chen.

Like many of you, we look forward to the conference and the discussion in our workshop in Tianjin.

Hope this report is useful to you, and perhaps it can serve as a starting point for our discussion. We would appreciate any feedback from you. Please also let me know if any information in Table 1 is incorrect. We shall write to you later about the record accuracy of Table 2.

ABSTRACT

The backward-facing step flow with an expansion ratio of 5 has been modelled by 19 teams without benchmark solution or experimental data. Different CFD codes, turbulence models, boundary conditions, numerical schemes and convergent criteria are adopted based on the participants' own experience in CFD simulation. The predicted non-dimensional penetration lengths as a function of Reynolds number are diverse among different teams. Even when the same turbulence model or even the laminar model is used, the difference is still notable among the results from different users. We believe that it indicates the combined effects of multiple decisions based on users' experience may cause significant difference in CFD "experiments". This calls for a solid approach of CFD validation and uncertainty assessment in CFD "experiments". A standard or guideline of using CFD and uncertainty assessment is needed to minimize the errors and uncertainties in the future.

INTRODUCTION

This is a preliminary summary of submissions from 19 teams worldwide for the CFD workshop in COBEE 2015.

Significant development and improvement has taken place in computational fluid dynamics (CFD) and its application in engineering, science and environment in the last 30 years. Many engineering fluid flow problems have been predicted and investigated by using CFD. It is known that due to the use of turbulence models, discretization schemes and limitations of user experience, errors can be introduced in CFD predictions. Errors due to choice of numerical methods, right physical model of the aimed engineering problem, relevant boundary conditions and errors due to user's experience are also a part of relevant issues in doing CFD predictions. The same is for predicting air flows in buildings.

It is well known that the low Reynolds' number effects may take place in many room air flows. This type of flow is difficult to predict by CFD due to limitations of available turbulence models in connections with RANS equations, limitations of computer capacity and also limitations in other types of models as e.g. the use of large eddy simulations (LES). Traditionally, we believe that two major sources of errors in CFD are related to two issues, i.e. solving the right equations, and the solving the equations rightly. Subsequently, various validation and verification methods have been developed to evaluate CFD in the two aspects. In general, we believe that as the mesh size and time step are sufficiently small, the computational solutions will approach to the exact solutions of the equations.

One relevant question to engineering application is how large is the uncertainty in CFD predictions for a particular engineering problem? Would the uncertainty vary for different flow problems? Which error (turbulence model, convection schemes or users) dominates? What need to be done to minimize such errors?

There have been a number of workshop studies in which organizers provide a benchmark solution or experimental data of one or more flow problems to the participants before or after the simulations (e.g. [Krogstad et al., 2013 and 2015](#)). Such studies have been useful in revealing the uncertainties and errors sources of CFD simulations in terms of turbulence models and discretization schemes, e.g. in combustion ([Lockwood et al, 2001](#)); hydrogen energy ([Baraldi et al., 2009](#)), and channel flows ([Rameshwaran et al., 2013](#)).

However, although the availability of experimental or benchmark data can be used to evaluate the CFD solutions, but it also introduces at least two problems – discouraging participation and influencing the users. Once the benchmark solution or experimental

data is provided, the participants could have more opportunities to validate their results then revise and improve the modelling. The variance of results may be underestimated.

Hence in the current exercises, the organizers of the workshop (Nielsen and Li) purposely decided that a fluid problem without any benchmark solutions is suggested. No participant including the organizers knows the answer. Our primary purpose is not in CFD validation and/or identifying which approach provides the most reliable prediction, but in exploring the potential differences if any in CFD predictions by different users, the reasons that may have caused the differences and how efforts can be made to minimize such differences. The closest and similar study to this one in the literature is probably Stewart et al (2012); however, a concurrent experimental study was carried out in Stewart et al (2012).

Since early 1990s, the issues of uncertainty of CFD predictions have been discussed, e.g. the pioneering work of Roache (1994). A review on CFD validation can be found in Stern et al (2006). Minimizing CFD uncertainty is crucial in many industrial applications, e.g. commercial aircraft design (Tinoco, 2008). A number of studies also focus on how to reduce uncertainty in CFD results, e.g. Mendenhall et al (2003). Study on variabilities of CFD solution exists like Steinman et al (2013). Quality assurance management was suggested for CFD verification and qualification (Colombo et al., 2012). Stern et al (2001) and Roach (1997), Ceili et al (2008), Oberkampf et al (2004) presented some of the few approaches widely used. The question is how to obtain “user-independent, mesh-independent and solver-independent” CFD solutions (Habashi et al., 2000).

There are many studies on CFD validation for improving the reliability of CFD predictions. The most relevant to building environment is that conducted by Srebric and Chen (2002). However, without any formal requirement, such a reporting requirement is seldom followed by industry. Among the limited number of studies on the impact of user experience on CFD use, the review paper of Johnson et al (2005) on the 30 years of development and application of CFD at Boeing is perhaps the most interesting to us. It finds that CFD “code must be very user oriented” allowing the ‘expert’ user to get fast results with reduced variation”.

In this study, we try to answer the following questions in this study:

- How large are the differences in the submitted CFD results by different users?
How significant are the differences? Why these differences exist? What can we do to minimize such differences?
- How the use of different turbulence models or discretization schemes impact on these differences?

Our observations and conclusions will be particularly useful to the users and modelers of CFD in the building and environment community, as most participating teams in our workshop are from this community.

METHODOLOGIES

The flow problem specification

The original idea of doing such a study was due to Nielsen, who proposed a simple CFD problem that is easy for all to simulate even on a personal computer for such as workshop study. It is incompressible and two-dimensional in the laminar regime and perhaps in the fully turbulent regime, but not necessarily in the transition regime. It may be considered as a simple building ventilation problem in the turbulent regime, and it is described as the backward-facing step or sudden expansion flow in the fluid dynamics community. The flow is isothermal.

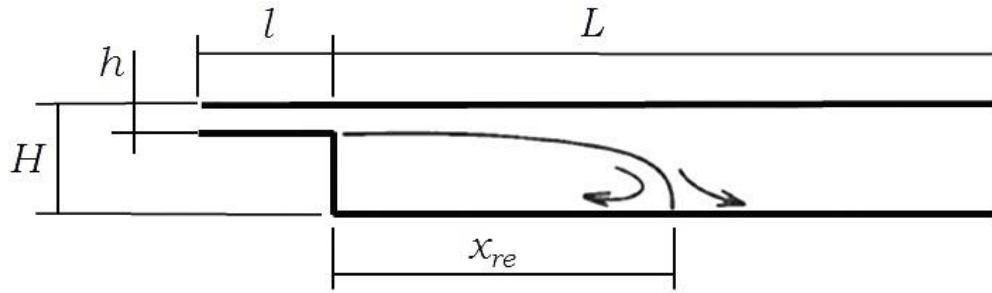


Figure 1. The geometry to use in the test case.

Figure 1 show the geometry of our proposed case. The flow is typical for isothermal room air flow in deep rooms. H , h and l are room height, supply slot height and length of supply opening, respectively. L is the length of the model/room, and x_{re} is the length from the end wall to the location where the reattaches flow is separated in a flow back to entrainment into the wall jet and a forward flow towards the exit (i.e. reattachment point). x_{re} is referred to as the penetration length of the supply jet.

Four methods (Le et al., 1997) can be used to determine the mean reattachment location, (a) by the location at which the mean velocity $U=0$ at the first grid point away from the floor; (b) by the location of zero wall-shear stress; (c) by the location of the mean dividing streamline; (d) by a p.d.f. method in which the mean reattachment point is indicated by the location of 50% forward flow fraction. The results of the first three methods are less than 0.1% of each other and about 2% different from the p.d.f result. Thus no matter which method the participants use, the results won't be influenced dramatically by this reason.

The following dimensions should be used:

$$h/H = 1/5$$

$$l/h = 4$$

L should have a sufficient length without influencing the obtained penetration length $x_{re}/(H-h)$. The L ranges from $4H$ to $40H$ in this study.

x_{re} is the distance to the first reattachment in the flow (there can be more in the case of laminar flow).

The flow might be transient and three-dimensional in a certain range of Reynolds Numbers (we can at least not exclude the possibility). We will therefore also define the test case as a 3D geometry with:

$$W = 2H$$

We consider the velocity and the penetration depth in the 3D case in the vertical median plane at:

$$y = 0.5W$$

The inlet flow is a top hat profile with a constant velocity u_o everywhere in the profile.

The Reynolds number is defined as:

$$Re = (h \cdot u_o) / \nu$$

where ν is kinematic viscosity.

Note that the backward-facing step flows have been considered as a good test case for flow separation and reattachment phenomena in fluid mechanics. The flow configuration is simple, but the flow behind the step is very complex, including the flapping phenomenon – the oscillations of the reattachment length of the primary circulation behind the step, and the large-scale vortical structures between the main flow and the recirculation regions, and these complex flow phenomena are yet to be fully understood (Schäfer et al 2009). Schäfer et al (2009) also provided a good literature review of the back-facing step flows. Many turbulence models have been applied for this flow, including direct numerical simulation (e.g. Le et al., 1997).

Biswas et al (2004) studied the impact of expansion ratios or three h/H ratios at 1/1.9423, 1/2.5 and 1/3. Experimental data are available in a number of studies, including one of the first studies (Armaly et al, 1983) and many others, e.g. Lee and Mateescu (1998). The present case of $h/H = 1/5$ is for an expansion ratio of 5, and it is unknown if any in-depth

studies have been carried out for such a large expansion ratio. However, such large expansion ratios are commonly found in building ventilation problems. Skovgaard and Peter (1991) studied the case of expansion of 6.

The availability of existing studies on the backward-facing step flows are not mentioned in our instruction to participants. The freedom is with any participating team to carry out any validation studies with any available data.

Invitation for participation

All participants are by invitation only, and they are all from active research teams in CFD for building airflows to the best knowledge of the workshops chairs, and/or recommended by other CFD experts.

The participants are asked to select a CFD code (commercial code or their own development) and they selected at least an appropriate turbulence model including LES. They are also free to select the boundary conditions. Hence effectively, the participants are acted like a potential CFD service providers to a client.

We also make arrangement so that each of the participants would be assigned a team number, C01, C02,...,C10... and all results be presented anonymously using the team number code, and only the particular participant and the coordinators know the code assignment, so any participant would not feel embarrassed in any way. The code is assigned randomly, and the code was sent to the modelers after the results were received.

In this arrangement, no one knows the exact results of the flow problem including us as coordinators.

Each team was asked to submit the predicted recirculation lengths for a minimum of 6 Reynolds numbers (between 1 and 10,000). The participants should also submit a very brief description of the simulations including CFD modeling strategies, the commercial codes, the grid, grid numbers, convergence criteria, number of iterations, turbulence model if used, and numerical schemes (convection schemes), and solution methods (SIMPLE, Multigrid) etc.

Submissions

A total of 22 teams agreed to participate in this workshop, and a total of 19 submissions were received by the deadline. Table 1 lists all of the participants who submitted the reports by team leaders' name, and Table 2 is a summary of simulation details with a random team code. Among these submissions, one team has used 3 CFD codes and the others used 1 code. The problem is considered as steady flow by most of the participants, and three teams (C09, C25 and C28) did unsteady analysis.

A total of 15 turbulence models are used, and most of the teams used 2 or 3 models, and one team chose to test a total of 9 models. The laminar model was used by all teams for the laminar case. The most widely-used turbulence models are the $k-\varepsilon$ family of turbulence models including the standard $k-\varepsilon$ model, the realizable $k-\varepsilon$ model, the RNG $k-\varepsilon$ model and the low Re $k-\varepsilon$ models. DES and LES are also utilized by some teams. In addition, the LVEL mode, a “heuristic” but useful model is adopted by one team. Algorithms such as SIMPLE, SIMPLEC, SIMPLEST and PISO scheme are used for pressure-velocity coupling, and the SIMPLE scheme is the most popular one. For the convection scheme, the majority of the participants chose the second order or the first order upwind scheme. In the 2D cases, the grid number ranges from 2,640 to 313,127, while in the 3D cases, the grid number ranges from 120,000 to 1,500,000. As shown in [Figure 2](#), the strategies of grid refinement are also diverse. Only Team C9 took a typical data set ([Lima et al., 2008](#); [Armaly et al., 1983](#)) for validation.

Table 1. List of participating teams

Team	Department/University/country	Team leader	Team members
1	School of Environment and Municipal Engineering/Xi'an University of Architecture and Technology/China	Angui Li	Li Gou
2	Shanghai University for Science and Technology/China	Haidong Wang	
3	School of Energy and Environment/Southeast University/China	Hua Qian	
4	1 School of Thermal Engineering/Shandong Jianzhu University/China 2 Department of Mechanical Engineering/University of Maryland, College Park/USA	Jelena Srebric ²	Jiying Liu ¹
5	Department of Mechanical and Aerospace Engineering/Syracuse University/USA	Jensen Zhang	Meng Kong
6	Department of Atmospheric Sciences/Sun Yat-sen University/China	Jian Hang	
7	Department of Building Services Engineering/The Hong Kong Polytechnic University/China	Jianlin Liu	Jianlei Niu
8	School of Environmental Science and Engineering/Tianjin University/China	Junjie Liu	Wenhua Chen, Zhuangbo Feng, Congcong Wang, Xingwang Zhao, Ying Zou, Fenghua Fan

9	Kato & Ooka Lab. , IIS/University of Tokyo/Japan	Li Wang	
10	School of Energy Science and Engineering/Central South University/China	Qihong Deng	Dan Mei, Ye Zhou
11	1 School of Mechanical Engineering/Purdue University, West Lafayette/USA 2 School of Environmental Science and Engineering/Tianjin University/China	Qingyan Chen ^{1,2}	Wei Liu ^{1,2}
12	1 Department of Civil Engineering/Aalborg University/Denmark 2 School of Architecture and the Built Environment, KTH Royal Institute of Technology/Sweden	Peter V. Nielsen ¹	Sasan Sadrizadeh ²
13	School of Civil Engineering/Dalian University of Technology/China	Tengfei Zhang	Shugang Wang, Jihong Wang
14	Department of Mechanical and Aerospace Engineering/Syracuse University/USA	Thong Dang	Mehmet Yildirim, Yang Zeng,
15	Department of Civil Engineering/Katholieke Universiteit Leuven/Belgium	Twan van Hooff	
16	Department of Building Science/Tsinghua University/China	Xianting Li	Yanqing Lin, Chao Liang, Huan Wang, Xiaoliang Shao
17	School of Environment and Municipal Engineering/Xi'an Univeristy of Architecture and Technology/China	Yi Wang	Yang Yang, Yu Zhou
18	Department of Mechanical Engineering/The University of Hong Kong/China	Yuguo Li	Lei Peng
19	Department of Mechanical Engineering/The University of Hong Kong/China	Yuguo Li	Han Yu

Table 2. List of the 19 teams, who submitted their predictions, as well as the associated software, turbulence models and numerical details.

Team code	CFD software	Models	Convection schemes	Grid	Solution methods	others
C1	STAR-CCM+ 2D	Laminar Realizable $k-\varepsilon$		313,127		20% TI L/H=15

C2	FLUENT 2D&3D	Laminar RNG $k-\epsilon$	2nd order upwind	2D:30,000	SIMPLE	5% TI L/H=20
				3D:120,000		
C3	FLUENT 2D	Laminar RNG $k-\epsilon$		178,000		5% TI L/H=11
C4	FLUENT 2D	Laminar Low Re $k-\epsilon$ (Abid)	1st order upwind	51,984	SIMPLE	L/H=16
		Standard $k-\epsilon$		44,344		
C6	FLUENT 2D	Laminar Standard $k-\epsilon$	P: Standard M: 2nd order upwind Others: 1st order upwind	31,436	SIMPLE	5% TI L/H=10
		S-A Standard $k-\omega$		64,241		
C7	FLUENT 3D	Laminar Realizable $k-\epsilon$	Second order upwind	548,640	SIMPLE	5% TI L/H=10
C9	FLUENT 2D	Realizable $k-\epsilon$	P: PRESTO! Convection: 1st order upwind	2,640		5% TI L/H=10
		v^2 -f model LES-WALE	P: PRESTO! M: bounded central differencing	24,030		
C11	PHOENICS 2D	LVEL	P: UPWIND, U,W: QUICK	13,440	SIMPLE	L/H=6.7
C12	FLUENT 3D	RNG $k-\epsilon$	1st Order Upwind	1,239,371	SIMPLE C	L/H=6.7
C13	FLUENT 2D	Realizable $k-\epsilon$	P: Standard, M, k, ϵ : 1st order upwind	71,111	SIMPLE	L/H=10
			P: Standard, M, k, ϵ : 2nd order upwind			
C14	FLUENT 3D	Laminar Realizable $k-\epsilon$ RSM		1,500,000	SIMPLE	L/H=10
C18	FLUENT 3D	Standard $k-\epsilon$	P: PRESTO!Others: 2nd order upwind	1,365,700	SIMPLE	L/H=24
C19	FLUENT 2D OpenFoam STAR- CCM+	Low Re $k-\epsilon$ (Abid) v^2 -f		51,200	SIMPLE	L/H=20
		Standard $k-\epsilon$ Realizable $k-\epsilon$ RNG $k-\epsilon$ SST, $k\omega$ -SST		28,160		L/H=15
C20	FLUENT 2D	Laminar Low Re $k-\epsilon$ (CHC)	2nd order upwind	18,240	SIMPLE	L/H=14.2
C25	FLUENT 3D	RNG $k-\epsilon$	2nd order upwind	992,000	SIMPLE C	L/H=12
		LES	bounded central		PISO	

		DES	difference			
C26	PHOENICS 3D	Low Re $k-\epsilon$ (LB) RNG $k-\epsilon$	Hybrid	320,000	SIMPLE ST	L/H=4
C27	STAR- CCM+ 2D	Low Re $k-\epsilon$ (AKN)	2nd order upwind	236,987	SIMPLE	L/H=40, TI = $0.16(\text{Re})^{-1/8}$
C28	STAR- CCM+ 2D, 3D	Laminar $k\omega$ -SST GR $k\omega$ -SST	2nd Order Upwind	200,000	SIMPLE	10% TI L/H=14
C29	FLUENT 3D	Laminar Realizable $k-\epsilon$		360,000	SIMPLE	10% TI L/H=10

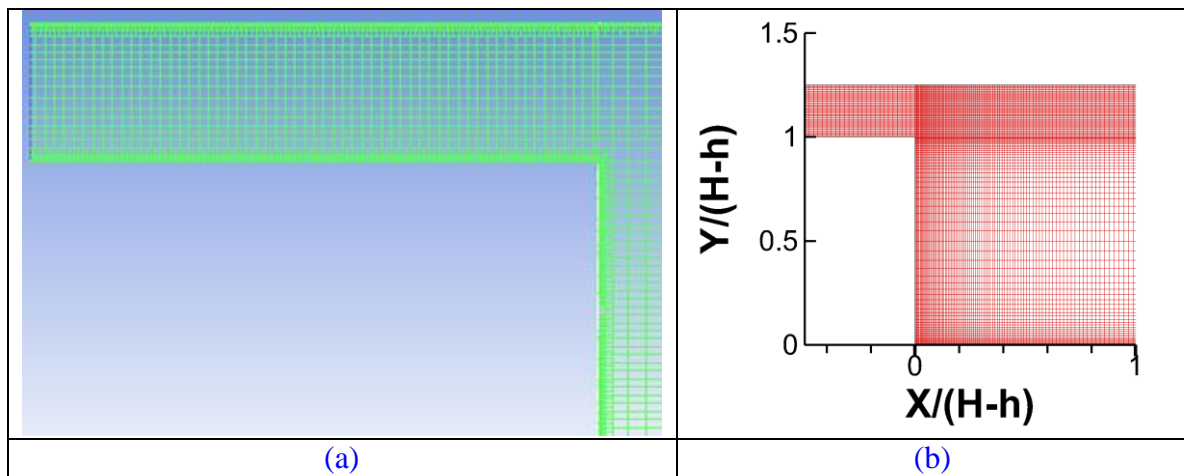


Figure 2. Grids around the inlet used by Team C6 (a) and that of Team C2 (b).

RESULTS

Significant difference in the prediction data by 19 teams

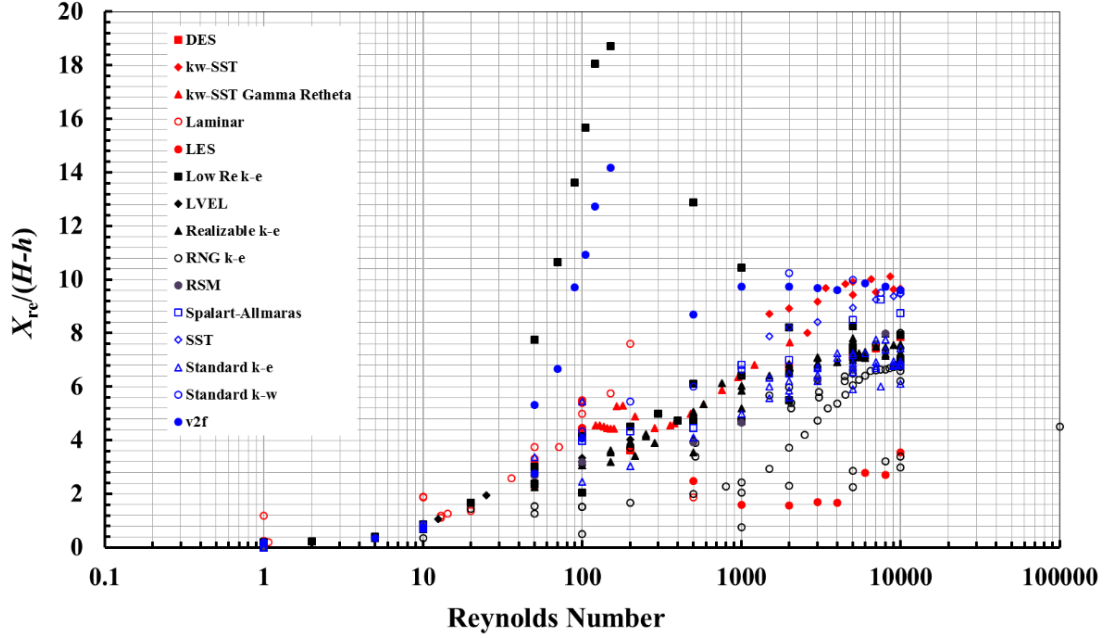


Figure 3. Predicted non-dimensional penetration length $\frac{X_{re}}{H-h}$ versus the Reynolds number of a total of all 362 data sets submitted by 19 teams

A total of 362 data sets are submitted by the 19 teams using 15 models. **Figure 3** shows the extremely diverse relationship between penetration length and Reynolds number predicted by all teams. The distribution of number of data sets for different Reynolds number range is shown in **Table 3**, including 168 data sets for the suspected laminar flow regime. The exact Reynolds number range for the regimes of laminar flow, transitional flow and turbulent flows are unknown.

With the increase of Reynolds number, the penetration length increases gradually for the Reynolds number less than 5000, then changes little when Reynolds number increases over 5000. Most of the penetration length values are less than 10, except those by a low Reynolds number $k-\epsilon$ model and v^2-f model. When the Reynolds number is larger than 1000, the predicted penetration length by the $k-\omega$ family of models is higher than the $k-\epsilon$ family of models. In addition, the penetration length predicted by the RNG $k-\epsilon$ model is smaller than the other RANS models for the same Reynolds number.

Table 3. The distribution of number of data sets for different Reynolds number range

Re range	(1, 500]	(500, 2000]	(2000, 5000)	[5000, 10000]	(10000, 100000]
Number of data sets	168	54	36	103	1

Predictions in the laminar regime

Most teams attempted a laminar model for this regime until the laminar model cannot produce a steady solution. A number of teams also attempted to use the turbulence model version for this regime. It is not safe to say that the flows for Re equal or less than 100 are laminar based on the submitted results, even though it may change to transitional flow before Re=100.

A total of 7 teams (C2, C3, C4, C6, C7, C14, C20, C29) used Fluent, and 2 teams (C1, C28) used STAR CCM+ to simulate the flow by laminar model with Reynolds number from 1 to 500. This model contributes most of the data for this regime. Figure 4 shows the relationship between non-dimensional penetration length and the Reynolds number by this model. With the increase of the Reynolds number, the reattachment length increases except when Reynolds number reaches to 500.

What is most surprising is that significant differences as large as 100% are found in the laminar flow predictions, this includes when the same software is used. For example, when Re=10 in Fluent 2D cases, the penetration length predicted by Team 03 is 0.75, while Team C6 and 20 predicted 1.9 and 1.88 respectively. When Re=100 in 3D cases, Team C7 and Team 29 predicted 4.33 and 5.39. The max difference is over 100% in Fluent 2D cases and over 20% in Fluent 3D cases.

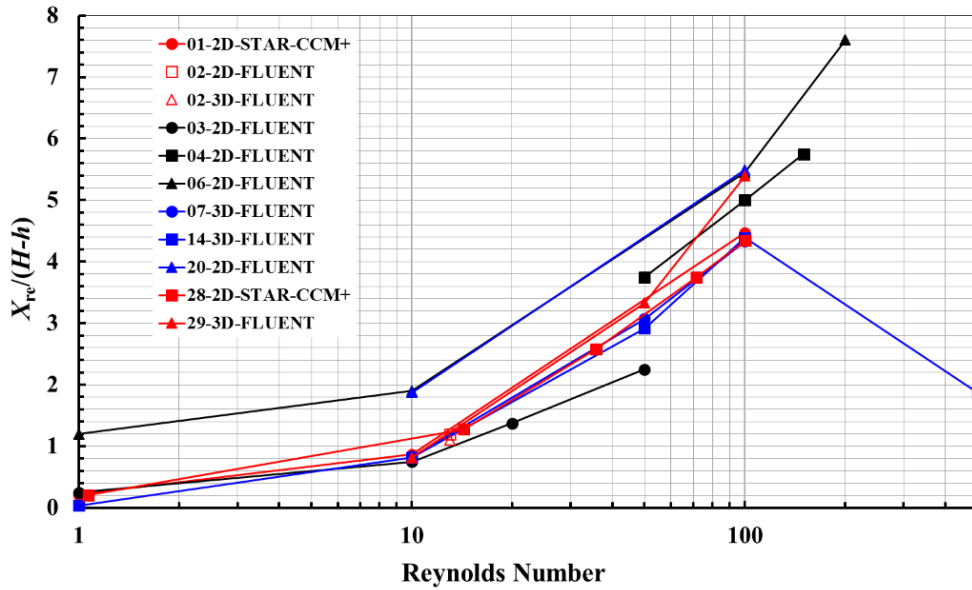


Figure 4. Relationship between non-dimensional penetration length $\frac{X_{re}}{H-h}$ and Reynolds number solved by laminar model.

The difference in prediction keeps in a certain level when the Reynolds number is lower than 100. As the Reynolds number is greater than 100, the differences become very significant, some studies even failed to obtain a converged solution (some might not have tried). Perhaps laminar flow is not a physical solution for $Re > 100$. Team C6 and C7 reported that they cannot obtain the convergent results when $Re=500$ using a laminar model, while Team C14 obtained the converged result. In these two cases, the velocity distributions are disorderly and unsystematic, and the residuals remain at a high level. In addition, the monitored values in these cases are unstable. So that the laminar model may not be suitable for the flow when Re is equal to or large than 500. Moreover, an obtained solution of the laminar equations is not a guarantee of laminar flow in reality. But without experimental data, we cannot make any conclusion.

Team C20 adopted a low Re $k-\epsilon$ model of [Chang et al. \(1995\)](#), developed for flows subjected to sudden pipe expansion, to simulated the airflow, he mentioned that the simulation for $Re=500$ required additional efforts to obtain a converged solution. Even though they reduce the under-relaxation factors, the convergence was still not completely perfect.

Some turbulence models such as v^2-f model, LVEL model, LES model and DES model, which can handle the low Reynolds number effect, are also adopted by some users. Figure 5 shows the results solved by these models. Team C25's results solved by LES and DES are almost the same. When Reynolds number is less than 200, all of the cases predicted similar penetration length, except for Team 19's v^2-f case. The difference begins to increase with the increase of Reynolds number at $Re>200$. We can conclude that the capability of these models is at same level in the laminar regime.

In addition, Team C28 added turbulence to the inlet conditions from 0 to 10 % and viscosity ratio $\mu_t/\mu = 10$. They show the solution is sensitive to the inlet turbulence intensity in a laminar case ($Re = 143$). This may be another reason why a large difference occurs in the laminar regime.

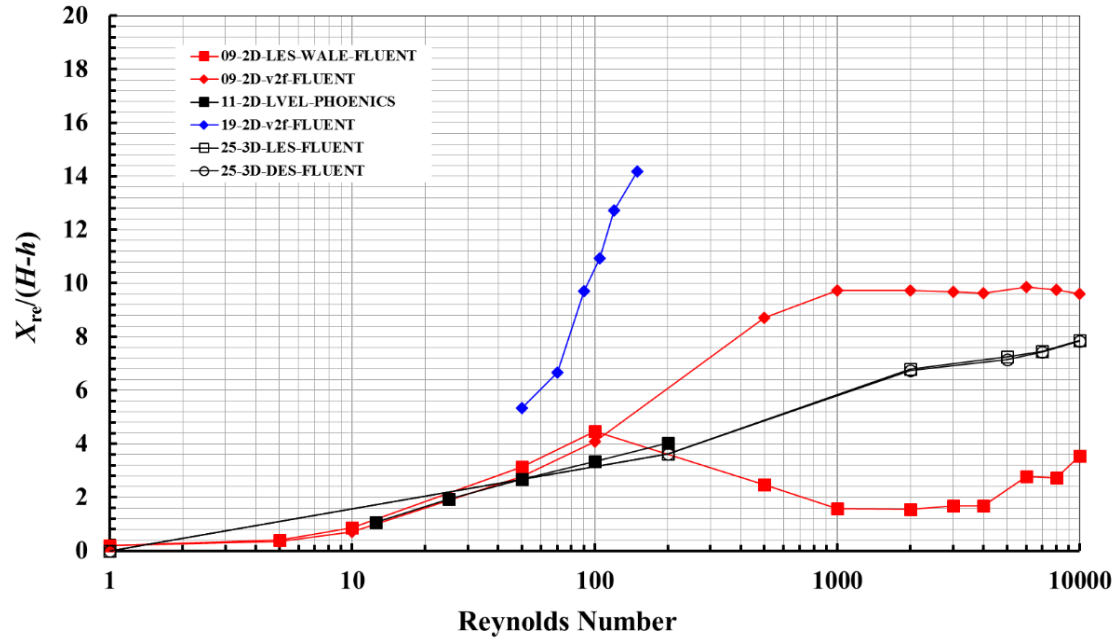


Figure 5. Relationship between non-dimensional penetration length $\frac{X_{re}}{H-h}$ and Reynolds number

The turbulence regime

Numerous turbulence models are developed to solve the turbulent flow in turbulence regime. When the Reynolds number is large than 5000, the slope of the penetration length versus the Reynolds number curve for each team approaches to zero in most of the cases. Hence we may consider that the flows for this range are fully turbulent for the sake of discussion here.

According to our literature review, numerous turbulence models provided by commercial CFD codes have been used to simulate fluid dynamics problems in turbulence regime, and some results are validated by experimental data, which proves the strong capability of the models. But it is hard to say which model is the best, because different model has their own advantages and disadvantages. Thus the difference caused by different turbulence models is not surprised. The preference of the turbulent models is normally determined by the users' own research experience or access to conclusion of other studies in the literature. In this workshop, the $k-\varepsilon$ family of models turns out to be the most popular among all users, which may reflect its popularity in industrial application. The most interesting finding is the results solved by the same model differ significantly from each other. Hence we shall discuss separately the differences arising use the same or different turbulence models.

The standard $k-\varepsilon$ Model

Four teams (C4, C6, C18 and C19) adopted the standard $k-\varepsilon$ model, most applied it when the Reynolds number is larger than 1000, and two teams also applied it with $Re \leq 1000$. The highest difference of penetration length occurs when Re is 100. This may due to the limitation of this model in the laminar regime. It is apparently the slope of penetration versus the Reynolds number curve is very small in each case. In the case of Team C19, three different CFD codes were used, and the recirculation length predicted by FLUENT is always smaller than the others. This means the CFD code may play an important role in affecting the CFD solution.

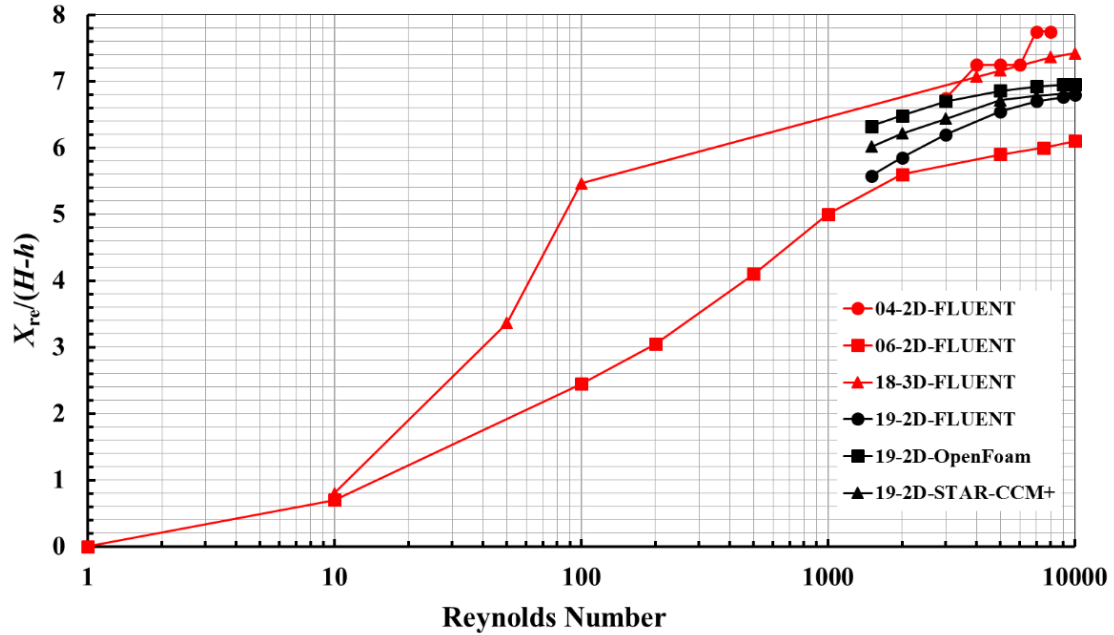


Figure 6. Relationship between non-dimensional penetration length $\frac{X_{re}}{H-h}$ and Reynolds number solved by the standard $k-\varepsilon$ model

The RNG $k-\varepsilon$ Model

Six teams (C2, C3, C12, C19, C25 and C26) adopted the RNG $k-\varepsilon$ model to conduct the simulation. Compared to the results of the laminar model and standard $k-\varepsilon$ model, the results solved by the RNG $k-\varepsilon$ model are extremely diverse and disciplined. Great difference not only occurs in the laminar regime, but also in the turbulent regime. Such significant difference proves the users are very important factor that influence the simulation results. For example, when $Re=10000$, we got 4 points. The highest and lowest predicted penetration length is 8 and 3 respectively, and the standard deviation of

4 points is 2.02. The different ratio is larger than 100% which is seldom see in the other blind tests.

As shown in Figure 7, the results of Team C2's results are different in terms of whether considering the problem as two dimensional or three dimensional. When Reynolds number is smaller than 4000, the predicted penetration length of 3D case is larger than the 2D cases, while the relationship reverses when Reynolds number is larger than 4000. According to Table 2, both Team C12 and 25 simulated the three dimensional flow by RNG $k-\varepsilon$ model with SIMPLER scheme and sufficient grid density. But their choices of convection scheme and convergence criteria are different. Team C12 used first order upwind and 10^{-3} , while Team 25 used second order upwind and 10^{-5} . From figure 7, we can find the difference of their results is huge, especially when Reynolds number is less than 5000. It predicts the choice is convection scheme and convergence criteria could lead to significant difference of CFD simulation.

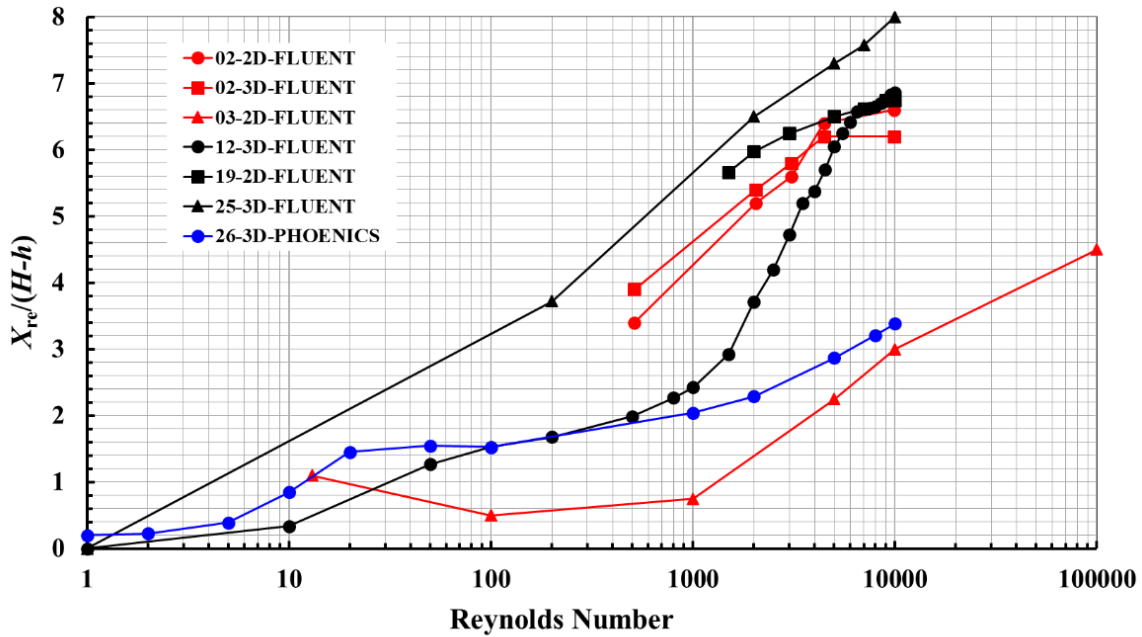


Figure 7. Relationship between non-dimensional penetration length $\frac{X_{re}}{H-h}$ and Reynolds number solved by RNG $k-\varepsilon$ model.

The realizable $k-\varepsilon$ Model

Seven teams (C1, C7, C9, C13, C14, C19 and C29) adopted the realizable $k-\varepsilon$ model to conduct the simulation. Compared to results of RNG $k-\varepsilon$ model, the difference between

different users is smaller, especially when Reynolds number is larger than 2000 or less than 200. The diversity focused on the transitional regime.

Team C13 used both first order upwind scheme (13A) and second order upwind wind scheme (13B) in their simulation, and the difference between two strategies is little. However, as discussed before, when Team C12 and 25 using different convection schemes with the RNG $k-\varepsilon$ model the difference is very significant. The reason is not clear yet.

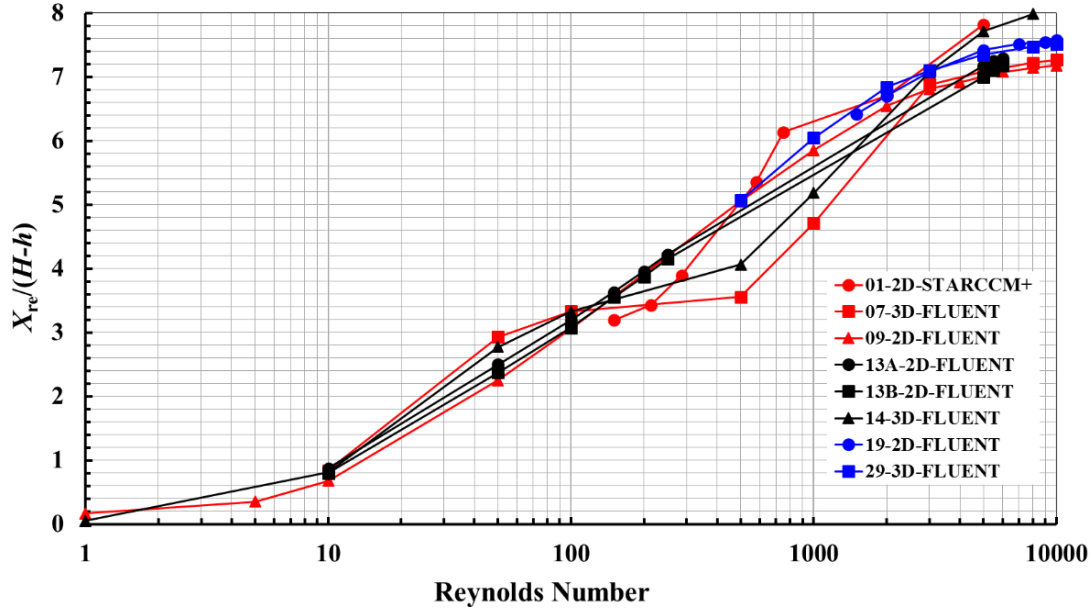


Figure 8. Relationship between non-dimensional penetration length and Reynolds number solved by Realizable $k-\varepsilon$ model.

The low Re $k-\varepsilon$ Models

A number of low Re $k-\varepsilon$ models are developed for the low Reynolds number effect, which can solve both the laminar flow and turbulent flow well. Considering the advantages of these models, five participants chose them to solve the problem. In this study, low re $k-\varepsilon$ models (C4: Abid, C19: Abid, C20: Chang-Hsieh-Chen, C26: Lam-Bremhorst, and C27: Abe-Kondoh-Nagano) are used. Unexpectedly, the solutions of different low Re $k-\varepsilon$ models are quite different. Compared with the results solved by other $k-\varepsilon$ models, the relationship between penetration length and Reynolds number is not positive correlation. Moreover, Adams and Johnston (1988) reviewed some experimental results with expansion ratio less than 3. It shows the physical penetration length is probably also not positive correlated with the Re number and a maximum penetration length in the beginning of the transient area.

Based on the limited data, the most significant difference appears at Re=100. The predicted penetration length by Team 26 is less than the others, it may due to the L/H ratio is only 4 in his case.

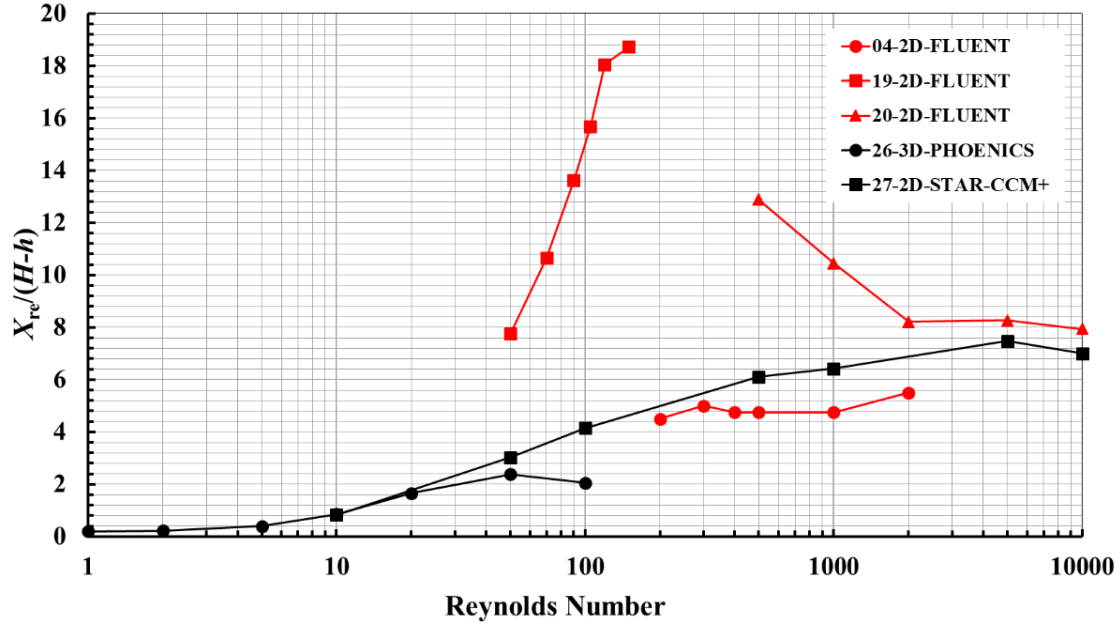


Figure 9. Relationship between non-dimensional penetration length $\frac{X_{re}}{H-h}$ and Reynolds number solved by low Re $k-\epsilon$ models. Note different low Re $k-\epsilon$ models were used by different teams; see text.

Influence of different models

The capacity of viscous model is another important factor that the CFD users should concern carefully. There is no doubt that an inappropriate choice of model would lead to significant error in CFD modelling. Here we choose Team C6's and Team C19's results to analyze the influence of models.

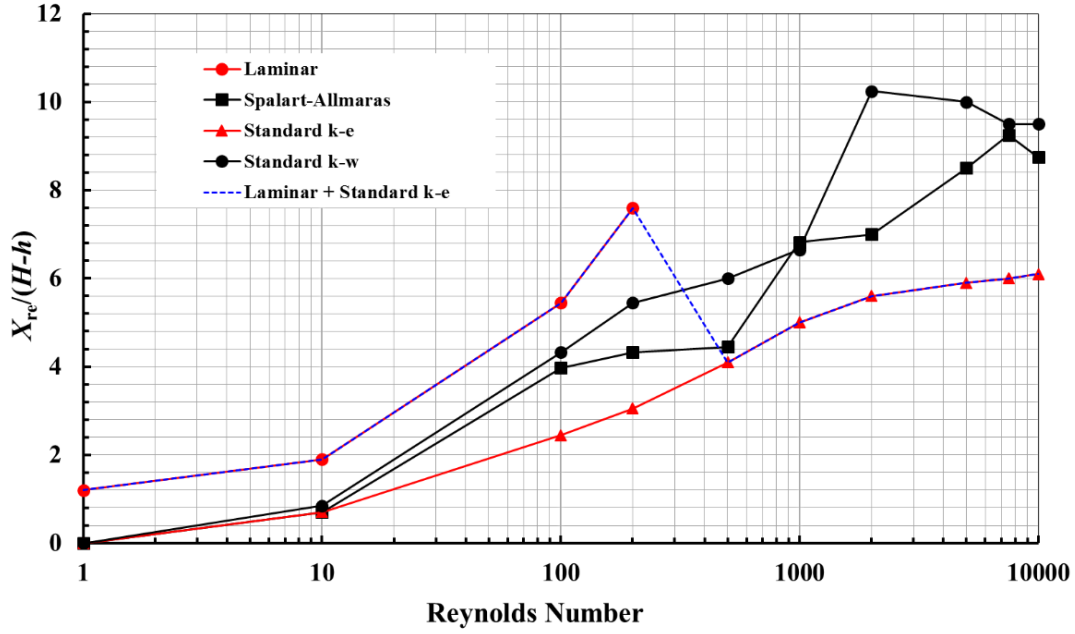


Figure 10. Relationship between non-dimensional penetration length $\frac{X_{re}}{H-h}$ and Reynolds number solved by Team C6.

Figure 10 represents Team C6's result solved by 4 different viscous models. For the laminar model and standard $k-\epsilon$ model, the total grid number is 31436, and the first grid near the wall is 5 cm away from the walls. For the Spalart-Allmaras model and the standard $k-\omega$ model, the grid used for CFD calculation is more intensive when compared to the grids used for $k-\epsilon$ model with standard wall function. The total grid number in these two models is 64251, and the first grid near the wall is about 1~2 cm away from the walls. The refined grid is shown in Figure 2(left). It is obvious that the penetration length solved by standard $k-\epsilon$ model is higher than other models in laminar regime. The result of the Spalart-Allmaras model may be quite abnormal. With the increase of Reynolds number, the penetration length increases stepwise but reduces when Re is higher than 2000.

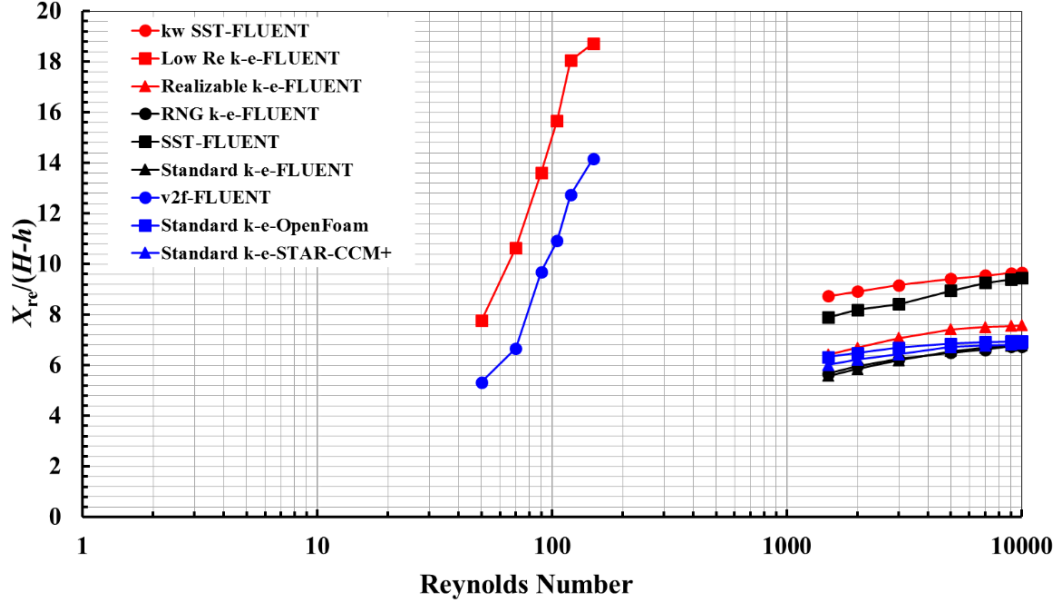


Figure 11. Relationship between non-dimensional penetration length $\frac{X_{re}}{H-h}$ and Reynolds number solved by Team C19.

Figure 11 represents Team C19's results solved by 7 models with 3 CFD codes. Because the low Re $k-\epsilon$ model and v^2-f model can simulate the low Reynolds number effect, and the two models are used when Reynolds number is equal or less than 150, and other models are used to solve the turbulent flow when Reynolds number is equal or larger than 1500. The standard $k-\epsilon$ model is adopted with Fluent, STAR CCM+ and OpenFoam, and other models are used with Fluent only. The grid is identical in all cases, and the numerical schemes are the same when Fluent is employed. Therefore, the difference is ascribed to the turbulence models. It is obviously that different codes also caused slightly difference with the standard $k-\epsilon$ model, but it is not notable as much as the difference caused by different models.

STAR-CCM+ 2D results

Most users utilized the Fluent, while some adopted STAR-CCM+. Figure 12 represents all of the results solved by STAR-CCM+. The difference in the predicted penetration length is very small in the laminar regime, but very notable in the turbulent regime. It includes the $k\omega$ -SST Gamma Retheta model, a transition model adopted by Team C28. There can be found a sudden increase of the penetration length when Re is about 180. In addition, they reported the penetration length is sensitive to inlet turbulence intensity, which is seldom noticed by CFD users. As shown in Figure 13, if the turbulence intensity decreases from 10% to 1%, the predicted streamline could change significantly. This effect could also be a reason that enlarges the difference of other viscous cases.

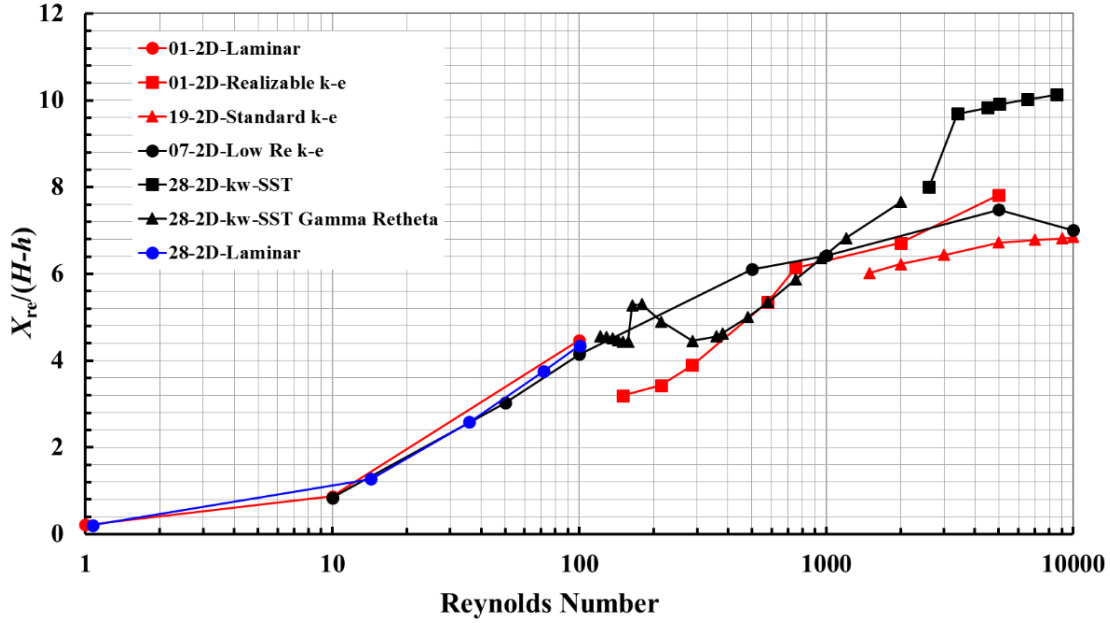


Figure 12. Relationship between non-dimensional penetration length $\frac{X_{re}}{H-h}$ and Reynolds number solved by STAR-CCM+.

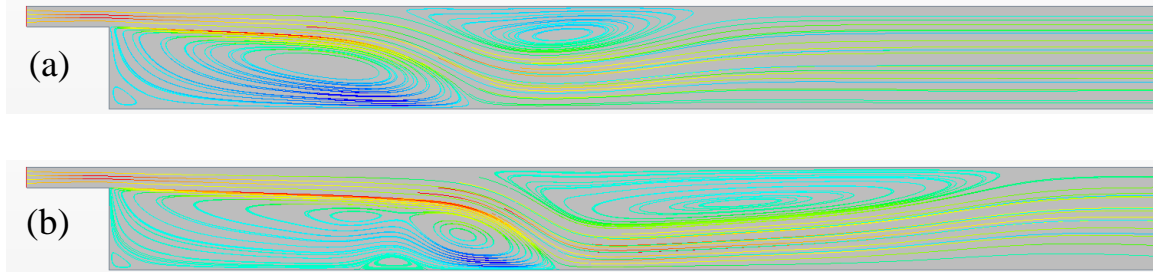


Figure 13. Streamline predicted by $k-\omega$ SST Gamma ReTheta with turbulence intensity of 10% (a) and 1% (b) (C28: Re=143)

Transitional regime

The flow in transitional regime is hard to be predicted theoretically or numerically, because the turbulence phenomenon is quite complicated. But the transitional effects always occur in indoor airflow, this problem cannot be evaded.

As mentioned before, two teams cannot obtain perfect converged solution with $Re = 500$ by the laminar model, as well as Team 20 who used a low Re $k-\epsilon$ model. It may due to the transitional effects starting to appear in this Reynolds number. When Reynolds number is equal to or larger than 5000, the penetration length is independent. Skovgaard

and Nielsen (1991) also found that there is a region in the transitional regime where the low Re k - ϵ model, as well as the high Re number version, fails to give converged results (for $h/H = 1/6$). Hence we assume Reynolds number from 500 to 5000 belongs to the transitional regime.

It is interesting that Team C25's DES and LES results are almost the same, but differ from Team 09's LES result.

DISCUSSION

Significant difference in the CFD results - the user difference may be the largest contributor for the significant difference

In this study, we use a blind test without benchmark solution or experimental to analyze the factors that could affect the CFD simulation, including the influence of turbulence models, the commercial codes, influence of boundary conditions, grid, simulation strategies and personal experience. Predicted penetration lengths by different models are diverse greatly. Moreover, the results solved by the same model and/or the same software could also have great difference, e.g. results of RNG k - ϵ model. Different convection schemes and convergence criteria could also lead to significant difference. According to Team C19's results, the CFD code also causes slightly differences. The inlet turbulence intensity is another important input parameter in modelling but seldom noticed by users. In addition, whether considering the flow as two dimensional or three dimensional will also lead to difference.

Unlike some other similar blind test with benchmark solution or reference data, the difference in our study is relatively more diverse. Since the all of the factors above are determined by the participants' personal experience, the user difference may be the largest contributor for the significant difference.

How to handle the transitional regime

In the last decades, the fluid mechanism in full turbulent regime and laminar regime is well studied in a certain level. Many of the turbulence models are deliberately developed for full-turbulent flow with high Reynolds number. Use of a turbulence model in the transitional regime is a particular worrying issue as there are not many suitable turbulence models for this region. In practice, for a CFD modeler, there is also no criterion for determining the existence of the transitional regime. Adams and Johnston (1988)'s review of previous experimental data shows the Reynolds number when peak penetration length appear could change with different expansion ratios. It indicates the boundary of transitional regime could change case by case. If a transitional flow model is chosen, when you switch to a different turbulence model is a problem.

The most popular k - ε family of turbulence models is capable of predicting the qualitative aspects of transition but sensitive to the initial conditions. Also, the beginning and end of transition are determined by the damping functions used in low Re k - ε models. They may be an important factor that made the results diverse. (Abid, 1993)

As shown above, the results solved by most popular turbulence models are extremely diverse. Even if the same models are used by different users, the difference could be found easily. The choice of turbulence models should be very carefully in CFD modelling. The boundary conditions, such as inlet turbulence intensity, should be specified carefully.

How to minimize user-dependent solutions?

Scientists repeat experiments to make sure the results can be replicated (Vaux, 2012 and Vaux et al., 2012). Based on this study, it seems to us that repeating CFD experiments may have become necessary and essential. There are many steps in doing a CFD simulation where users may introduce errors, just like doing a physical experiment. We may recommend repeating CFD experiments as doing physical experiments. For the laminar case, it is obviously that there is probably one correct solution, but due to user errors most likely, multiple solutions are resulted. In analog with physical experiments, experiments may be repeated on the same experimental set up by the same researcher(s), or on a new set up, or by a different research group. If we interpret a CFD simulation as a CFD “experiment”, repeating the CFD simulation may be done from scratch by the same user or by a different user, or by using different software. Testing, retesting, checking and confirming CFD experiments are needed to reduce errors and uncertainties.

Simulations should also start with relevant validations. All CFD prediction should follow the relevant established guidelines, and in the case of built environment applications. However, though these guidelines are helpful in minimizing the errors due to the choice of grids, turbulence models, boundary conditions and numerical methods. The many other decisions made by the users are also important, such as setting fluid properties, convergence criteria, and all those decisions not covered in validation.

The uncertainties which are connected to CFD predictions are sometimes handled by requiring benchmark tests of situations closely similar to the actual case to be studied. This will help the user to optimize his conditions around the prediction, like using the right equations, an effective turbulence model and other important decisions, and therefore getting closer to the actual situation. For example smoke management calculations are often supplied with CFD predictions of similar benchmarks to support the fire and smoke calculations.

Due to the significant uncertainty produced due to the users' experience, there may be a need for an ISO standard, similar to the 1993 ISO Guide to the expression of uncertainty in measurement.

CONCLUSIONS

The backward-facing step flow has been comprehensively modelled by 19 teams without benchmark solution or experimental data. A wide variety of turbulence models, including RANS, DES and LES approaches, were employed based on the participants' experience. The results are more diverse than expected. The predicted penetration length not only differs due to the use of different viscous models, but also from different users when the same model was used. Different numerical schemes and convergent criteria can lead to significant difference sometimes. Choice of different codes also contributes to difference of solutions.

Each aspect above in CFD simulation could lead to a notable difference, and the combined effects of multiple decisions based on users' experience may cause significant differences. Proper computer simulation design as done for physical experiments (e.g. Montgomery 2013) is meaningful.

REFERENCES

- Abid, Ridha. "Evaluation of two-equation turbulence models for predicting transitional flows." *International Journal of Engineering Science* 31, no. 6 (1993): 831-840.
- Adams, E. W., & Johnston, J. P. (1988). Effects of the separating shear layer on the reattachment flow structure part 2: Reattachment length and wall shear stress. *Experiments in Fluids*, 6(7), 493-499.
- Armaly, Bassem F., et al. "Experimental and theoretical investigation of backward-facing step flow." *Journal of Fluid Mechanics* 127 (1983): 473-496.
- Baraldi, D., Kotchourko, A., Lelyakin, A., Yanez, J., Middha, P., Hansen, O. R., ... & Molkov, V. (2009). An inter-comparison exercise on CFD model capabilities to simulate hydrogen deflagrations in a tunnel. *international journal of hydrogen energy*, 34(18), 7862-7872.
- Biswas, G., M. Breuer, and F. Durst. "Backward-facing step flows for various expansion ratios at low and moderate Reynolds numbers." *Journal of fluids engineering* 126.3 (2004): 362-374.
- Celik, I. B., Ghia, U., & Roache, P. J. (2008). Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *Journal of fluids Engineering-Transactions of the ASME*, 130(7).

- Chang, K.C., Hsieh, W.D., Chen, C.S., (1995) A modified low-Reynolds-number turbulence model applicable to recirculating flow in pipe expansion, *Journal of Fluids Engineering*, 117, 417–423.
- Coleman, H. W., & Steele, W. G. (2009). *Experimentation, validation, and uncertainty analysis for engineers*. John Wiley & Sons.
- Colombo, E., Inzoli, F., & Mereu, R. (2012). A methodology for qualifying industrial CFD: The Q 3 approach and the role of a protocol. *Computers & Fluids*, 54, 56-66.
- Franke, J., Hellsten, A., Schlünzen, H., & Carissimo, B. (2007). Best practice guideline for the CFD simulation of flows in the urban environment.
- Gallego, E., Garcia, J., Migoya, E., Crespo, A., Kotchourko, A., Yanez, J., ... & Molkov, V. (2005, September). An intercomparison exercise on the capabilities of CFD models to predict deflagration of a large-scale H₂-Air mixture in open atmosphere. In *Proceedings of International Conference on Hydrogen Safety*, Pisa, Italy.
- Habashi, W. G., Dompierre, J., Bourgault, Y., Ait - Ali - Yahia, D., Fortin, M., & Vallet, M. G. (2000). Anisotropic mesh adaptation: Towards user - independent, mesh - independent and solver - independent CFD. Part I: General principles. *International Journal for Numerical Methods in Fluids*, 32(6), 725-744.
- Johnson, F. T., Tinoco, E. N., & Yu, N. J. (2005). Thirty years of development and application of CFD at Boeing Commercial Airplanes, Seattle. *Computers & Fluids*, 34(10), 1115-1151.
- Krogstad, Per-Åge, and Pål Egil Eriksen. "“Blind test” calculations of the performance and wake development for a model wind turbine." *Renewable energy* 50 (2013): 325-333.
- Krogstad, Per-Åge, Lars Sætran, and Muyiwa Samuel Adaramola. "“Blind Test 3” calculations of the performance and wake development behind two in-line and offset model wind turbines." *Journal of Fluids and Structures* 52 (2015): 65-80.
- Le, Hung, Parviz Moin, and John Kim. "Direct numerical simulation of turbulent flow over a backward-facing step." *Journal of fluid mechanics* 330 (1997): 349-374.
- Lee, T., and D. Mateescu. "Experimental and numerical investigation of 2-D backward-facing step flow." *Journal of Fluids and Structures* 12.6 (1998): 703-716.
- Lima, R. C., Andrade, C. R., & Zaparoli, E. L. (2008). Numerical study of three recirculation zones in the unilateral sudden expansion flow. *International Communications in Heat and Mass Transfer*, 35(9), 1053-1060.
- Lockwood, F. C., Abbas, T., Kandamby, N. H., & Sakthitharan, V. (2001). CFD experience on industrial combustors. *Progress in Computational Fluid Dynamics, an International Journal*, 1(1-3), 1-13.

- Mendenhall, M. R., Childs, R. E., & Morrison, J. H. (2003, January). Best practices for reduction of uncertainty in CFD results. In 41st AIAA Aerospace Sciences Meeting.
- Montgomery, D (2013). Design and analysis of experiments (8th ed.). Hoboken, NJ: John Wiley & Sons, Inc.
- Oberkampf, W. L., Trucano, T. G., & Hirsch, C. (2004). Verification, validation, and predictive capability in computational engineering and physics. *Applied Mechanics Reviews*, 57(5), 345-384.
- Rameshwaran, P., Naden, P., Wilson, C. A., Malki, R., Shukla, D. R., & Shiono, K. (2013). Inter-comparison and validation of computational fluid dynamics codes in two-stage meandering channel flows. *Applied Mathematical Modelling*, 37(20), 8652-8672.
- Roache, P. J. (1997). Quantification of uncertainty in computational fluid dynamics. *Annual Review of Fluid Mechanics*, 29(1), 123-160.
- Schäfer, F., M. Breuer, and F. Durst. "The dynamics of the transitional flow over a backward-facing step." *Journal of Fluid Mechanics* 623 (2009): 85-119.
- Skovgaard, M., and Nielsen, P. V. (1991). Numerical Investigation of Transitional Flow over a Backward Facing Step Using a Low Reynolds Number $k-\varepsilon$ Model. Aalborg: Dept. of Building Technology and Structural Engineering. (Indoor Environmental Technology; No. 22, Vol. R9150).
- Srebric, J., & Chen, Q. (2002). An example of verification, validation, and reporting of indoor environment CFD analyses (RP-1133). *TRANSACTIONS-AMERICAN SOCIETY OF HEATING REFRIGERATING AND AIR CONDITIONING ENGINEERS*, 108(2), 185-194.
- Steinman, D. A., Hoi, Y., Fahy, P., Morris, L., Walsh, M. T., Aristokleous, N., ... & Habets, D. F. (2013). Variability of computational fluid dynamics solutions for pressure and flow in a giant aneurysm: the ASME 2012 Summer Bioengineering Conference CFD Challenge. *Journal of biomechanical engineering*, 135(2), 021016.
- Stern, F., Wilson, R. V., Coleman, H. W., & Paterson, E. G. (2001). Comprehensive approach to verification and validation of CFD simulations—part 1: methodology and procedures. *Journal of fluids engineering*, 123(4), 793-802.
- Stern, F., Wilson, R., & Shao, J. (2006). Quantitative V&V of CFD simulations and certification of CFD codes. *International journal for numerical methods in fluids*, 50(11), 1335-1355.
- Stewart, S. F., Paterson, E. G., Burgreen, G. W., Hariharan, P., Giarra, M., Reddy, V., ... & Malinauskas, R. A. (2012). Assessment of CFD performance in simulations of an idealized medical device: results of FDA's first computational interlaboratory study. *Cardiovascular Engineering and Technology*, 3(2), 139-160.

- Tilley, N., Rauwoens, P., & Merci, B. (2011). Verification of the accuracy of CFD simulations in small-scale tunnel and atrium fire configurations. *Fire Safety Journal*, 46(4), 186-193.
- Tinoco, E. N. (2008). Validation and minimizing CFD uncertainty for commercial aircraft applications. *AIAA paper*, 6902.
- Vaux, D. L. (2012). Research methods: know when your numbers are significant. *Nature*, 492(7428), 180-181.
- Vaux, D. L., Fidler, F., & Cumming, G. (2012). Replicates and repeats—what is the difference and is it significant?. *EMBO reports*, 13(4), 291-296.